Numerical Simulation of a Displacement Ventilation System with Multi-heat Sources and Analysis of Influential Factors

Xuan Wu
Doctor
The key laboratory of clean coal power generation and combustion technology of the ministry of education, southeast university
Nanjing, China
Wuxuan5110@sina.com

Jingfang Gao
Instructor
College of energy sources & environment, Inner Mongolia University of Science & Technology
Baotou, China
soucegjf@sina.com

Wenfei Wu
Professor
College of energy sources & environment, Inner Mongolia University of Science & Technology
Baotou, China
wwf@imust.cn

Abstract: Displacement ventilation (DV) is a promising ventilation concept due to its high ventilation efficiency. In this paper, the application of the CFD method, the velocity and temperature fields of three-dimensional displacement ventilation systems with double heat sources are numerically simulated. The model is verified by experimental data.

The results of the study show that thermal stratification characteristics exist in indoor temperature fields. The paper also analyzes the influence of different influential factors, e.g., the distance between heat sources, temperature of heat source, heat characteristics of the wall and outdoor temperature. It was found that the human requirement for comfort is satisfied easily when the distance between heat sources is long. Under the conditions simulated in this paper, when the distance was more than 0.8m, the temperature distribution tended to be average and steady, and it did not change as the distance changed. Second, the temperature change of the thermal current has a large influence on the indoor temperature. The rise in thermal current temperature makes the vertical temperature gradient in the room increase. The upper temperature of the room becomes higher, as does the height of the high temperature air level that lies in the upper part of the room. Finally, both the heat loss of the surrounding structure and the change in outdoor temperature have a large influence on indoor temperature. However, it does not influence the thermal stratification characteristics of DV. The only thing that has changed is the thermal stratification height.

Key words: displacement ventilation；heat comfort；double heat sources；numerical simulation；CFD

1. INTRODUCTION

With the gradual progress in human society, people more and more concern the quality of indoor air. Although the traditional mixing systems have poor ventilation efficiency and are less energy efficient, they still occupy a large portion of the market. When displacement ventilation (DV) was first introduced almost three decades ago, it seemed at the time to be a promising ventilation concept due to its high ventilation efficiency and stratification principle. However, few research work is engaged in displacement ventilation simulation with multi heat sources [1]–[3]. Only some experimental studies are carried out [4]－[5]. In practice, DV system is a complex system that many pollution heat sources dynamic exist simultaneously. In this mixed system, there exist interaction among pollution heat sources and limitation of thermodynamic concentration diffusion among different pollution air. So, it’s greatly significant to engage in research of DV with multi heat sources. In this paper, distribution of velocity and temperature in displacement ventilation system are investigated by using computational fluid dynamics (CFD).

2. METHODS

Three-dimensional geometry (3.0m×3.0m×3.0m) under consideration is shown in Fig. 1. A primary pollution heat source (0.4m×0.4m) is located at the bottom of room and an accessory heat source (0.1m×0.4m) located nearby. A return air outlet is installed near a ceiling in the opposite wall side. In this study, seven representative dots (A, B C, D, E, F and G) which distributed around heat source are selected to analyze the whole temperature distribution in room.

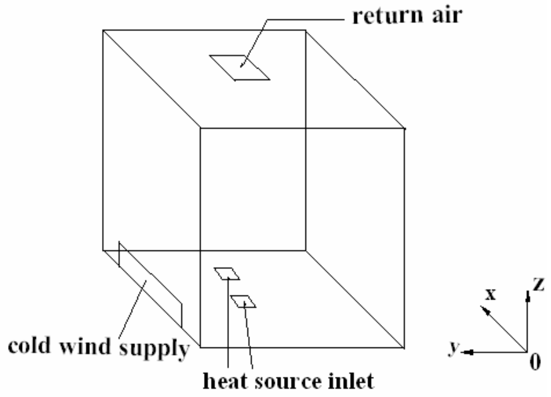


Fig. 1 Model of simulated room

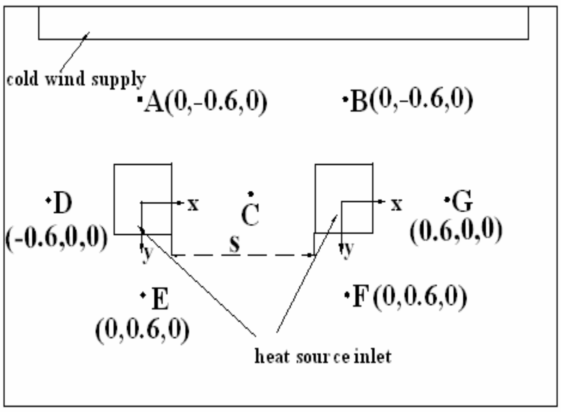


Fig. 2 Distribution of four dots

The finite volume method is employed to solve the time-averaged Navier-Stokes equations with a collocated variable arrangement. The SIP solver is used to solve the set of algebraic equations. A conventional turbulent $k-\varepsilon$ model with wall functions is adopted. The semi-implicit method for pressure-linked equations (SIMPLE) is used to reach a convergent solution set. The CFD program is written in FORTRAN language. The calculated velocity and temperature fields are visualized by TECPLOT9.0.

It is assumed that flow is steady, turbulent, Newtonian and incompressible with constant physical properties. The continuity, momentum, energy conservation and $k-\varepsilon$ equations are expressed as

$$\frac{\partial}{\partial x_j}u_j = 0 \tag{1}$$

$$\rho\frac{\partial}{\partial x_j}u_i u_j = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial}{\partial x_j}\rho(u_i' u_j') + F_i \tag{2}$$

$$\rho C_P\frac{\partial}{\partial x_j}Tu_j = \frac{\partial}{\partial x_j}\left(-k\frac{\partial T}{\partial x_j}\right) + \frac{\partial}{\partial x_j}\rho C_P(u_j' T') + \Phi \tag{3}$$

$$\rho\frac{\partial k}{\partial t} + \rho u_j\frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\eta + \frac{\eta_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \eta_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \rho\varepsilon \tag{4}$$

$$\rho\frac{\partial \varepsilon}{\partial t} + \rho u_j\frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\eta + \frac{\eta_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{c_1\varepsilon}{k}\eta_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \frac{c_2\rho\varepsilon^2}{k} \tag{5}$$

Where u_i, T, and p are the mean components, and u_j', T', and p' are fluctuation components of an instantaneous velocity, temperature and pressure respectively. $\rho(u_i'u_j')$, $\rho C_P(u_j'T')$ in Eqs.(2) and (3) represent additional stress(Reynolds stress) to a fluid element and additional heat flux due to turbulent phenomena. Φ is a mean dissipation term which is not considered in this study due to low velocity scale. F_i is the ith component of buoyant force due to temperature difference. Eqs. (4) and (5) is called the $k-\varepsilon$ model. In this model, the eddy viscosity is expressed as

$$\eta_t = c_\mu \rho k^2 / \varepsilon \tag{6}$$

This model contain five parameters, the most commonly used values are

$$C_\mu = 0.09;\ C_1 = 1.44;\ C_2 = 1.92;\ \sigma_k = 1.0;\ \sigma_\varepsilon = 1.3 \tag{7}$$

In this study, the Boussinesq approximation is appropriate for the simulation. In region near the wall, no-slip boundary condition and the wall functions are used. The radiative heat transfer of wall is neglected. The wall is adiabatic.

Because of the effect of natural convection, a kind of rising thermal current with certain speed will be formed above the solid heat source. At the same time, in order to analyze the unique air distribution of the DV system as well as its characteristics to restrain the polluted sources and to prevent its diffusion, we approximately apply the hot air-supply opening to replace the solid heat source in real project, and use to simulate the hot rising stream above the solid heat sources [6]. The approximation may result in certain difference between the simulation and the real project, but this kind of simplification won't have great influence on the study of the characteristics of DV systems. That is to say, the difference between the simulation results and the real condition will be small.

3. RESULTS

The model is verified by experimental data [1]. Fig.1 shows the experimental room (1.8m×1.2m×1.5m) has two heat pollution heat sources ($S = 0.4m$). Supply air enters from a wall-mounted, low-velocity diffuser with velocity of 0.11m/s and temperature of 24.1℃. The temperature of heat sources is 48.5℃. Fig.3 shows the trend of temperature change of dot A along the height (0.125m-1.125m) of the room. It is observed that simulation results are almost consistent with experimental data. The maximum relative error is 2%, which proves that the mathematical model built in this paper is right and the hypothesis is reasonable

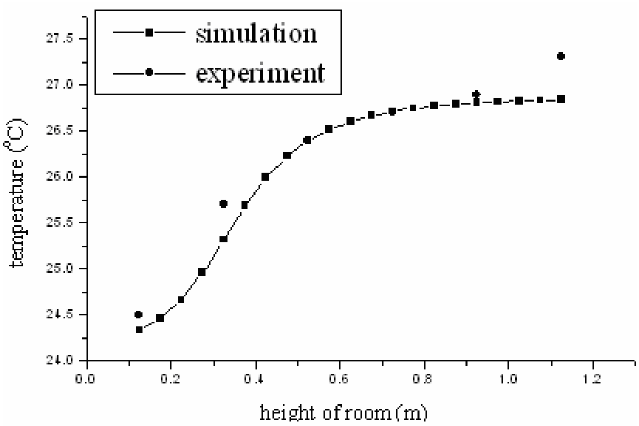


Fig.3 Comparison of experiment and simulation

Firstly, whether people can stay between two heat sources, and how the change of distance influences on the comfort of indoor people. In order to make it clear, The DV system with four different conditions ($S = 0.4m, 0.6m, 0.8m$ or $1m$) is simulated. We choose its central position C as the observation point.

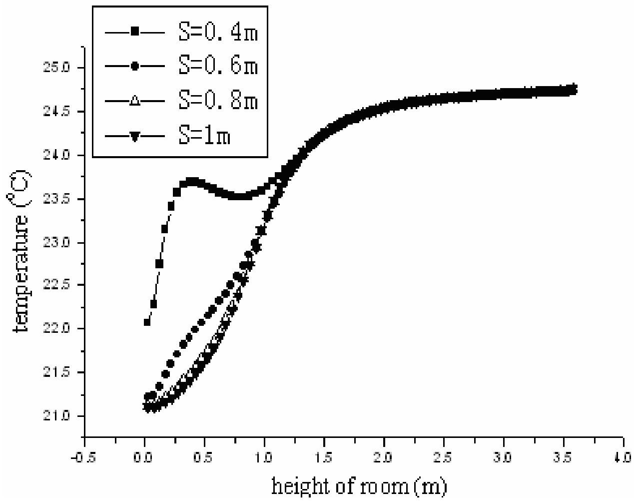


Fig.4 Influence of the distance

Fig.4 shows that when the distance S is 0.4m, the temperature distribution between two heat sources was most influenced by the diffusion of the heat sources, in the lower part of the room. The reason is that the short distance between heat sources. Consequently, only small amount of cold wind can get into the area in which it can not restrain the diffusion of the thermal current effectively. Obviously, it will be dangerous for people to stay in at that time. As the distance between multi-heat sources increases, the temperature in the lower part of point C became lower and lower. This change tends to be uniformity. Under the condition that the paper simulates, it will be comfortable for people to stay in when the distance between heat sources is bigger than 0.6m. Besides, the temperature in the upper field of point C hardly changed as the distance between heat sources increased again. Through study above, we should pay more attention to the influence that the change of distance between two heat sources on the vertical temperature between them.

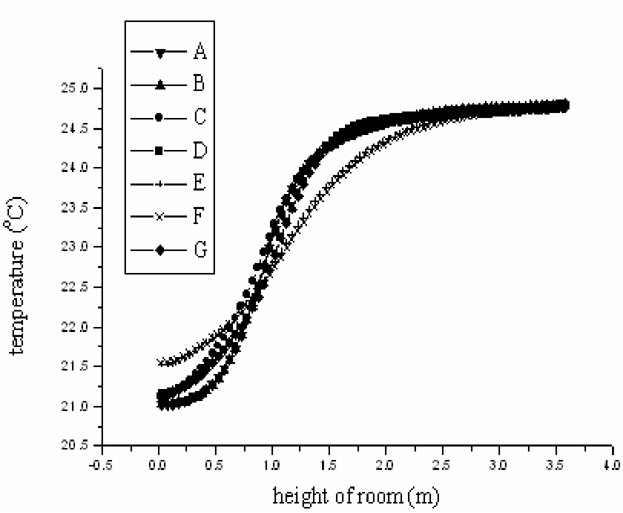
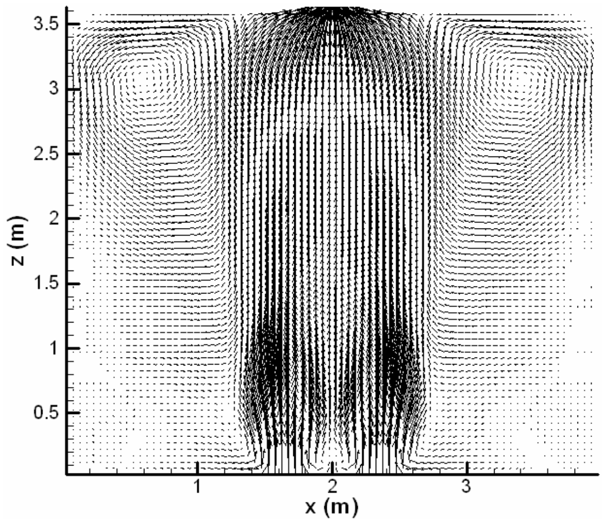


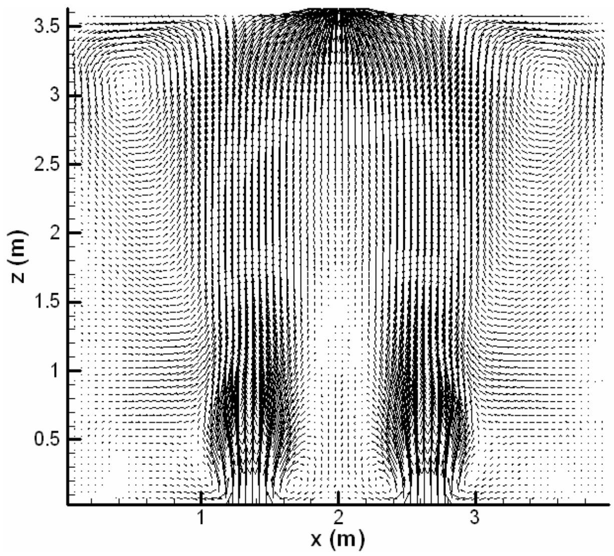
Fig.5 Vertical temperature field ($S = 0.8m$)

Fig.5 shows the vertical temperature distribution at seven points around the two main heat sources when the distance between them is 0.8m. The thermal stratification characteristic exists in the temperature field indoors. The temperature in the lower part of the room was very low, while the temperature gradient was high, so it was considered as a layer of low temperature air. On the contrary, the upper part of the room was considered as a layer of high temperature air, because the condition was just the opposite. And because of the decline of supply air speed as well as the obstruction from the heat sources, the vertical temperature distribution of point E and point F is a little different from that of other points.

Fig.6 shows the velocity distribution in x-z section when the distance between heat sources S is 0.4m or 1m. When $S = 0.4m$, there is only little cold air that can get inside the two heat sources, so it can not restrain the diffusion of thermal current effectively. But when $S = 1m$, the situation is contrary. The change of the velocity field determines the change of the temperature field.



a) x-z section ($S = 0.4m$)



b) x-z section ($S = 1m$)

Fig.6 Velocity distribution of DV with multi-heat sources.

We choose two different conditions when the supply air temperature of heat sources was 40℃ and 43℃ to have a contrast. We assume that the distance between heat sources was 1m, and other parameters keep unchanged. From Fig.5 we can see that the temperature at each point is nearly the same, so here we just choose the vertical temperature change at point A to describe the temperature of the whole field.

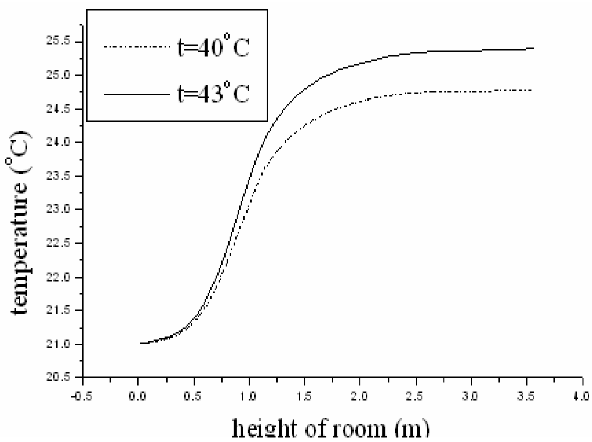


Fig.7 Influence of heat sources temperature

Fig.7 shows the comparison of vertical temperature distribution at point A under the above two conditions. It shows the vertical temperature gradient in the room and the temperature in the upper part of the room become higher and higher, as the supply air temperature of the heat sources increases. It is because the rising of the thermal current temperature makes the heat load in the room increases. At that moment, the temperature difference between the heights (0.1m-1.1m) is above 3℃. Obviously, it can not satisfy people's requirement for comfort, so it need to supply more wind to eliminate the increased indoor heat load. But on the other hand, that might be limited by the room condition and people's comfort. Therefore, at this time, we can not simply depend on the DV system to eliminate the indoor load efficiently and to satisfy people's requirement for comfort. So we need to add a refrigeration system. At present, the best program is to combine the DV system with the chilled ceiling system which uses the cold radiation to transmit heat.

Finally, we analyze the influence that the heat loss of surround structure on displacement ventilation under two conditions: one is considered as project A: the surround structure is all formed by heat wall. Another project B: the heat transfer coefficient of the wall on the opposite of the air-supply opening is 0.72 $w/m^2\cdot{}^oC$, and other walls are adiabatic. Here we just assume the outdoor temperature as a constant 30℃, it will not change with time. Under both of the two conditions, the supply air temperature of heat sources is 40℃, and the distance between two heat sources is 100cm, which other parameters keep unchanged. As to the second condition, the outside wall can be dealt with as the third boundary condition.

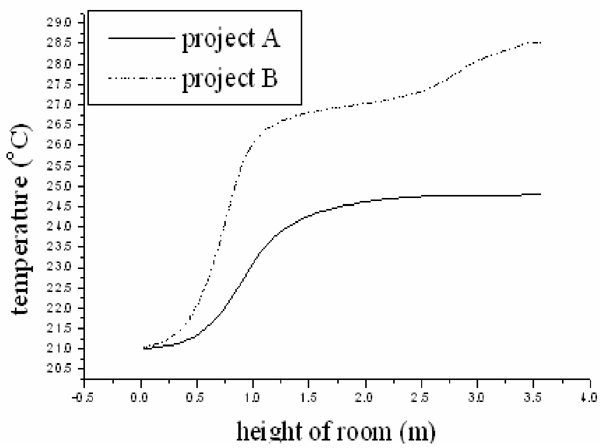


Fig.8 Influence of surrounding structure

Fig.8 shows the comparison of the vertical temperature distribution at point A under these two conditions. From it we can learn that the indoor temperature field has greatly different under the two conditions, it is because the outdoor temperature is higher than the indoor temperature, the indoor heat load is enhanced. It results in great increase of the indoor temperature and the temperature gradient, and also has great influence on the height of the high temperature air floor and low temperature air floor of the indoor temperature field. Obviously, we need more cold wind to counteract the increased heat load and to satisfy people's requirement for comfort.

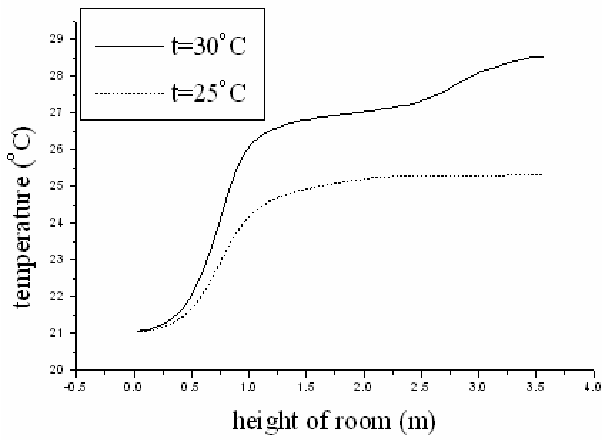


Fig.9 Influence of outdoor temperature

Next, we analyze the influence that the change of outdoor temperature on DV system. In order to avoid the influence of other factors, we still assume that the outdoor temperature will not change with time. We choose two conditions when the outdoor temperature was 30℃ and 25 ℃ to analyze.

Fig.9 shows the change of the outdoor temperature had great influence on the indoor temperature. To be specific, the rising of outdoor temperature not only make the temperature gradient of low temperature air layer and the temperature in the lower part of the room increased, also makes temperature gradient of high temperature air layer and the temperature in the upper part of the room rise. That is same with the result from experiment in [7]. The reason lies in the different outdoor temperature, it lead to different surface temperature of the inside wall, and lead to different amount of thermal load that get into the room through the wall, and therefore have a great influence on the indoor temperature.

4. CONCLUSION

Numerical simulation of the DV system which has main and assistant heat sources is carried out in this paper. We can get the following conclusions:

People’s requirement for comfort can be satisfied easily when the distance between heat sources is long. Under the condition simulated in this paper, when the distance was more than 0.8m, the temperature distribution tend to be average and steady, and it will not change as the distance changing. At the same time, the thermal stratification characteristic exists in the temperature field indoors.

The temperature of the thermal current has great influence on the indoor temperature. The rising of the thermal current temperature makes the vertical temperature gradient in the room increase. The upper temperature of the room becomes higher, and also the height of the high temperature air level which lies in the upper part of the room rise.

Both the heat loss of surrounding structure and the change of outdoor temperature have great influence on the indoor temperature. But it will not influence the thermal stratification characteristic of displacement ventilation. Only one has changed is the thermal stratification height.

REFERENCES

[1] Hee-Jin Park, Dale Holland. The effect of location of a convective heat source on displacement ventilation: CFD study[J]. Building and environment.2001.36. 883-889.

[2] Ma Guobin. Air flow organization of displacement ventilation and analytic study of influential factors[J]. Building Energy & Environment. 2002;3.5-12. (In Chinese)

[3] H.Xing, A.Hatton, H.B.Awbi. A study of the air quality in the breathing zone in a room with displacement ventilation[J]. Building and environment. 2001.36. 809-820.

[4] Junjun Chen. The experiment research of displacement ventilation about many polluting heat sources[D]. Master Paper. Xian university of construction technology. 2003. (In Chinese)

[5] Ni Bo. Experimental study of displacement ventilation system with two heat sources[J]. HV&AC. 2002; 32(5).17-19. (In Chinese)

[6] D.Blay. Convective phenomena involved in a displacement ventilation system[C]. Indoor Air’96, 1996, Part I:1-10.

[7] Ni Bo. Experiment of indoor temperature and velocity field for displacement ventilation [J]. HV&AC. 2000; 30(5). 2-5. (In Chinese)